



National Defence
Research and
Development Branch

Défense nationale
Bureau de recherche
et développement

TECHNICAL MEMORANDUM 96/232

February 1997

HOT SPOT STRESS ANALYSIS OF A
FATIGUE SPECIMEN USING
FINITE ELEMENT METHOD

Thomas S.Z. Hu

19970402 027

NOT REPRODUCED 4

Defence
Research
Establishment
Atlantic



Centre de
Recherches pour la
Défense
Atlantique

Canada

DISSEMINATION STATEMENT R
Approved for public release
Distribution Unlimited

DEFENCE RESEARCH ESTABLISHMENT ATLANTIC

9 GROVE STREET

P.O. BOX 1012
DARTMOUTH, N.S.
B2Y 3Z7

TELEPHONE
(902) 426-3100

CENTRE DE RECHERCHES POUR LA DÉFENSE ATLANTIQUE

9 GROVE STREET

C.P. BOX 1012
DARTMOUTH, N.É.
B2Y 3Z7

Mar-59



National Defence
Research and
Development Branch

Défense nationale
Bureau de recherche
et développement

HOT SPOT STRESS ANALYSIS OF A FATIGUE SPECIMEN USING FINITE ELEMENT METHOD

Thomas S.Z. Hu

February 1997

Approved by R.W. Graham:
Head/Hydroautics Section

TECHNICAL MEMORANDUM 96/232

Defence
Research
Establishment
Atlantic



Centre de
Recherches pour la
Défense
Atlantique

Canada

Abstract

DREA is a member of the International Ship & Offshore Structure Congress (ISSC) and currently participates in two of the technical committees (II.1 and III.1). It is recognized that the accuracy of the stress results of a finite element analysis has significant effect in the fatigue life prediction of the structure. Participants of the committee II.1 (the quasi static load effects) have agreed to perform a finite element comparative study. The emphasis of the study is on hot spot stress analysis in which the stress results can be used to determine the fatigue life and the stress concentration factor. In order to draw a quantitative conclusion, a stiffened plate, tested in Japan, was selected to be the subject of this study. Participants were asked to use their own solution procedures and modelling techniques independently. After presenting the background information on hot spot stress analysis in fatigue life prediction and a brief description of the specimen, this memorandum summarizes the finite element model and the solution procedure used by DREA. Numerical results obtained with the finite element method and analytical results from simple beam theory are presented and discussed.

Résumé

Le CRDA est membre de l'International Ship & Offshore Structure Congress (ISSC) et est actuellement présent dans deux comités techniques (II.1 et III.1). Il est reconnu que la précision des résultats concernant les efforts calculés par une analyse par éléments finis est très importante lors de la prédiction de la durée de vie d'une structure. Les participants du comité II.1 (au sujet des effets des chargements quasi-statiques) se sont mis d'accord pour réaliser une étude comparative en utilisant la méthode des éléments finis. Pour cette étude l'accent a été mis sur l'analyse des efforts hot spot stress dont les résultats peuvent être utilisés pour déterminer la durée de vie et les facteurs de concentration des efforts. Afin d'obtenir des résultats quantitatifs plutôt que qualitatifs, une plaque renforcée testée au Japon a été choisie comme cas pratique de l'étude. On a demandé aux participants d'utiliser leur propre démarche et les techniques de modélisation qui leur sont propres de façon indépendante. Après une présentation générale des connaissances concernant la prédiction de la durée de vie d'une structure par analyse des efforts hot spot stress et la description du cas test, cette note technique présente succinctement le modèle par éléments finis et la démarche adoptée par le CRDA pour résoudre ce problème. On présente et on discute les résultats numériques obtenus avec la méthode des éléments finis ainsi que les résultats analytiques issus de la simple théorie des poutres.

Hot Spot Stress Analysis of a Fatigue Specimen Using Finite Element Method

by

Thomas Hu

Executive Summary

Introduction

Fatigue cracking is a common threat to ship structural integrity. The fatigue life of a structural component can be predicted with the hot spot stress approach using finite element methods. However, the results of the finite element analysis of a particular problem may vary with the solution procedures and the judgement of the analysts. As a member of the International Ship & Offshore Structure Congress (ISSC), DREA develops new ship structural and constructional knowledge through cooperation with other countries. One of the current cooperative activities is a finite element comparative study. It is hoped that these exercises may result in some finite element guidelines on how and where to perform the analysis in the fatigue life prediction. All participants in one of the latest studies were asked to perform a hot spot stress analysis by using the finite element method to model a stiffened plate tested in Japan. With the experimental results, a more meaningful quantitative conclusion about the numerical methods may be drawn.

Principal Results

This memorandum describes the background information on the use of hot spot stress analysis in fatigue life prediction. After a brief description of the specimen, this memorandum outlines the finite element model and solution procedure and summarizes the results. The commercial finite element program ALGOR was chosen for this exercise. In order to easily compare the stress results with the test measurements, the finite element mesh was generated with an automatic mesh generator so that all finite element nodal points were coincident with the strain gauge locations and points of interest. Since the test results had not been released at this time, the finite element results were compared with those calculated with simple beam theory. The effective breadth concept was used to explain the differences.

Significance of Results

It is concluded that finite element methods can be used not only to predict the fatigue life but also to study the suitability of the detailed design of the brackets. In order to obtain a meaningful result, however, users should understand not only the fatigue life phenomena but also the finite element formulations and their limitations.

Future Plans

A future analysis using the DREA program VAST to solve the same problem is planned. The results can be used to verify the accuracy and compare the solution time. The preprocessing and postprocessing capability of VAST can also be verified and any deficiencies identified.

Contents

Abstract	ii
Executive Summary	iii
Table of Contents	iv
List of Figures	v
List of Tables	vi
Nomenclature	vi
1. Introduction	1
2. Hot Spot Stress in Fatigue Analysis	3
3. Description of the Problem	5
4. Finite Element Model	10
5. Numerical and Analytical Results and Discussion	12
6. Concluding Remarks	26
References	27

List of Figures

1. Test set-up	6
2. Specimen geometry	7
3. Details of the brackets and collar plate (in millimeters)	8
4. Locations of the strain gauges	9
5. Finite element model	11
6. σ_{xx} across the plate at location $X=461$ mm	17
7. σ_{xx} across the plate at location $X=1200$ mm	17
8. σ_{xx} across the web of the side beam at location near the weld toe	18
9. σ_{xx} across the web of the central beam at location near the weld toe	18
10. σ_{xx} across the web of the central beam at location $X=1500$ mm	19
11. σ_{xx} across the web of the central beam at location $X=1900$ mm	19
12. Displacements along the flange of the central beam	20
13. Displacements along the flange of the side beam	21
14. σ_{xx} of the central beam at the farthest fibres from the neutral axis	22
15. σ_{xx} of the side beam at the farthest fibres from the neutral axis	23
16. σ_{xx} at the vicinity of the bracket of the side stiffener	24
17. Deformed shape	25
18. Von Mises stress distribution	25

List of Tables

1: Stresses at various strain gauge locations	14
2: Stresses at locations near the weld toe of the side beam	15
3: Hot spot stresses predicted by various rules	16

Nomenclature

E	Young's modulus
I	moment of inertia
L	length of the beam
M	moment
P	load
R_x, R_y, R_z	rotational displacement degrees of freedom about x, y and z axes
U_x, U_y, U_z	translational displacement degrees of freedom in the x, y and z directions
x	distance from one end of the beam
y	distance from the neutral axis of the cross section
σ, σ_{xx}	stress parallel to the longitudinal axis
δ_x	displacement at location x

1. Introduction

DREA is a member of the International Ship & Offshore Structures Congress (ISSC) and currently participates in two of the technical committees: Committee II.I - the quasi-static load effects and Committee III.I - the ductile collapse. It is recognized that the results of the finite element analysis of a particular problem may vary with the solution procedures and the judgement of the analysts. A finite element comparative study was conducted by members of the committee II.I in the previous congress (ISSC'94) [1]. Nine committee members and two outside collaborators contributed to this study. The study was conducted independently by participants with their own solution procedures. A side structure of a middle size crude oil carrier was selected to be the subject. The study was divided into two phases, a global analysis of the side structure; and the detailed evaluation of the local stresses at the intersection of a longitudinal stiffener and a transverse frame.

The results of the global analysis showed that the stresses were strongly affected by the element types and mesh subdivisions. In the local stress analysis, most of the participants used a zooming procedure to acquire the detailed model from the global structure. This procedure required different hypotheses such as boundary displacements and tractions to be introduced into the model. As expected, the results also showed large variations. Because of the lack of experimental evidence, no quantitative conclusions could be drawn. Obviously, some specific guidelines on how and where to perform the stress analysis are necessary. This is particularly important when the stress results are used in fatigue life prediction, since the predicted fatigue life may be changed by several hundred per cent by a small stress variation.

Since the determination of the stress is so important in the prediction of the fatigue life, the mandate of Committee II.I for the next congress (ISSC'97) has been changed to emphasize the importance of the stress prediction in the numerical methods:

"Concern for the quasi-static response of ship and offshore structures, as required for safety and serviceability assessments. Attention shall be given to uncertainty of calculation models for use in reliability methods, and to the combination of exact and approximate methods, for the determination of stresses appropriate for different acceptance criteria. "

In the last interim meeting [2], participants of Committee II.I have agreed to perform another finite element comparative study. To further investigate the effect of the finite element modelling on the stress results and to provide insight on the consequences for the fatigue life prediction, the committee decided that the problem should be fatigue related and should have existing experimental evidence. Therefore, a finite element comparative study on the hot spot stress prediction, for which the results can be used to determine the fatigue life and stress concentration factor (SCF), was outlined. One of the fatigue tests done in Japan [3] was selected to be the subject of the investigation.

The background information on the use of the hot spot stress in the fatigue analysis is reviewed in Section 2. A brief description of the specimen including the geometry, the test setup and the strain gauge locations is reported in Section 3. The solution procedures including the finite element program, the element type and description of the numerical model are outlined in Section 4, while the analytical results and discussions are summarized in Section 5.

2. Hot Spot Stress in Fatigue Analysis

From ship structural integrity and the constructional procedure considerations, the longitudinal stiffeners are required to be welded to the transverse bulkheads via brackets. These weldments create notches while the improper details in the design and fabrication of the brackets can result in areas of stress concentrations.

A study [4] done on ten tankers has indicated that 40 % of the cracks have occurred in connections of side shell longitudinals to transverse bulkheads or web frames. These fractures occur almost exclusively at the weld toes. Elastic finite element analysis has shown that the stresses at most of these cracked locations are much less than yield. It points to fatigue as the crack initiation mechanism.

Since longitudinal stiffeners are subjected to repeated tensile and compressive stresses under wave loading, these components are much more susceptible to fatigue cracking. A crack initiates at the area of stress concentration. It can propagate through the longitudinal stiffener until it reaches the shell plate. Subsequently, it creates a surface fracture that goes through the plating. If this crack is left unrepaired, it can grow to a critical length and result in quick crack growth. As a result, it may create a significant environmental impact and large financial losses.

There are three separate but interconnected fatigue cracking stages i.e. crack initiation, crack growth and rapid crack growth (fracture). The first two can be predicted by different approaches. Crack initiation is usually analyzed by using stress exceedance or S-N curves which describe the relationship between stress range and number of cycles to failure. Crack growth can be analyzed by using fracture mechanics technology, which relates experimental crack growth data to stress concentration factors (SCFs). Both require the stress information near the crack tip.

The S-N curves used in most fatigue design standards are based on experimental data obtained from constant amplitude tests on particular weld details. In addition to the shape of the detail, a weldment creates notches including changing section due to the reinforcement of the weld, surface ripples, lack of penetration, undercut, shrinkage cracks, lack of fusion, porosity, and inclusions. These curves incorporate both the notch effect of the weld and the stress concentration due to the detail. According to their shapes, these details are classified or catalogued in various standards [5,6].

The fatigue life is usually expressed in terms of the stress in the vicinity of the joint. Structural designers classify the weld joint of interest into one of the classes or catalogues and select the appropriate S-N curve. According to the desired service life, they can determine the allowable stress range. In some cases, however, the class of the weld joint cannot be identified easily. The hot spot stress approach provides an alternative.

The use of the hot spot stress approach allows a wide variety of welded configurations to be considered on a common basis. With sophisticated numerical methods, the highly detailed stress distribution including the stress concentration effects of the detail can be calculated. Because the calculated stresses include the effects of stress concentration due to the detail, the designed S-N curves only need to include the notch effect of the weld itself. The calculated stress information can, therefore, be plugged into one of the simplest S-N curves, such as that for transverse groove welds (no effects of the stress concentration), to determine the fatigue life. The hot spot stress approach will gradually become more popular as detailed stress analysis using finite element methods becomes a more common numerical tool.

The American Petroleum Institute (API RP 2A-LRFD) [7] defines hot spot stress as the stress in the immediate vicinity of a structural discontinuity. Due to the steep stress gradients near a discontinuity, stresses at strain gauges further away from the weld may be too low. The hot spot strain gauge is typically placed within a very short distance (6 mm) from the weld toe and oriented perpendicular to the weld.

According to the API specification, for tubular members and connections exposed to variations of stress due to environmental or operational loads, either the American Welding Society (AWS) [5] X or X' curve should be used. The mathematical representation of the X curve is

$$N = 2 \times 10^6 \left(\frac{\Delta \sigma}{\Delta \sigma_{ref}} \right)^{-m}$$

where N=number of cycles; m=inverse log-log slope (4.38 for the X curve); $\Delta \sigma$ =applied stress range; and $\Delta \sigma_{ref}$ = stress range at reference point (100MPa at 2 million cycles for the X curve). The microscopic notch at the weld toe is included in the empirical X curve provided the as-welded profile merges smoothly with the adjoining base metal. For a weld without such profile control, the X' curve (modified from the X curve) is applicable [8].

API suggests that the X and X' curves be used with hot spot stress ranges based on suitable SCFs. SCFs may be derived from finite element analyses. The API code further says that finite element analyses of thin shells may be used to obtain hot spot strains compatible with the X curve, provided the mesh is fine enough to reproduce the steep stress gradients and the results correspond to the actual welded toe location, rather than the mid-plane intersection. Isoparametric thick shell and solid elements can be used for even more direct modelling of the weld zone.

3. Description of the Problem

The details of the test specimen can be found in reference [3]. A brief description of the test specimen and the requirements of the analysis are presented here.

The test specimen is a full scale model representing the typical side longitudinal and transverse bulkhead intersections of the crude oil tankers. It is made of high strength steel having a yield stress of 314 MPa. A three-point bending moment is applied through a hydraulic actuator as shown in Figure 1. Load is gradually increased from zero to the maximum of 392kN. The detailed geometry of stiffeners, plates and brackets are shown in Figures 2 and 3. The strain gauge locations are marked in Figure 4.

The comparison study concentrated on the following items:

- Deflection at the center of the model - point D in Figure 1;
- Stresses at the major measuring points as indicated in Figure 4:
 - E19, E20, E21, E22, E23, E24, E25, E33 and E34 for the central longitudinal;
 - E37, E38, E39, E44, E45, E46, E47, E55 and E56 for the side longitudinal;
- Stresses at the flange of the side longitudinal near the weld toe of the bracket:
 - E39, E40, E41, E42, E43 and other points which will be used to extrapolate the hot spot stress.

In order to compare the numerical results, all the participants are asked to use the same parameters:

- Type of the analysis - linear elastic analysis;
- Material properties - linear elastic isotropic material with Young's modulus equal to 206 GPa and Poisson's ratio equal to 0.3;
- Load magnitude - 392 kN;
- Stress and strain components to be reported - in the direction parallel to the longitudinals.

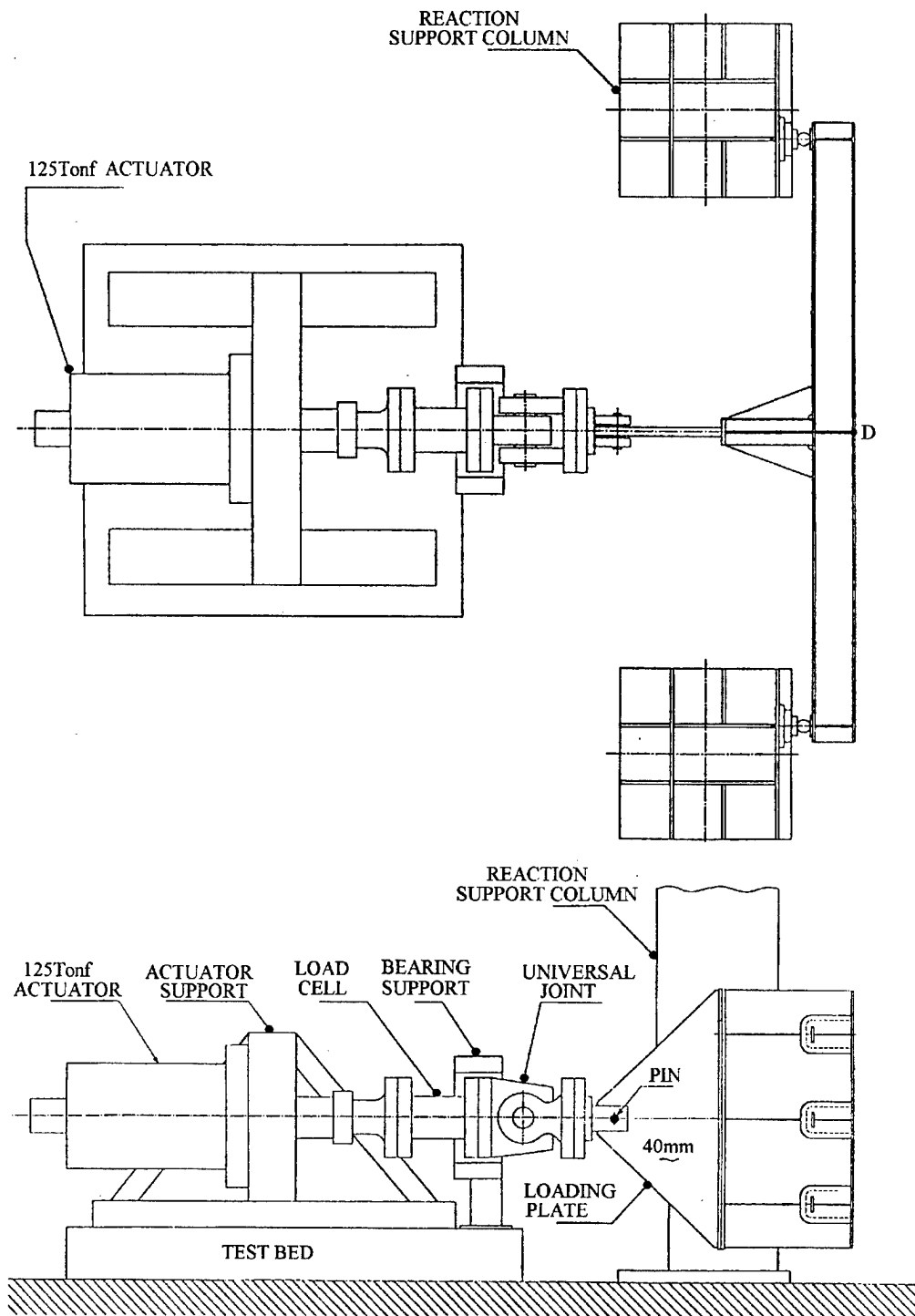


Figure 1: Test set-up

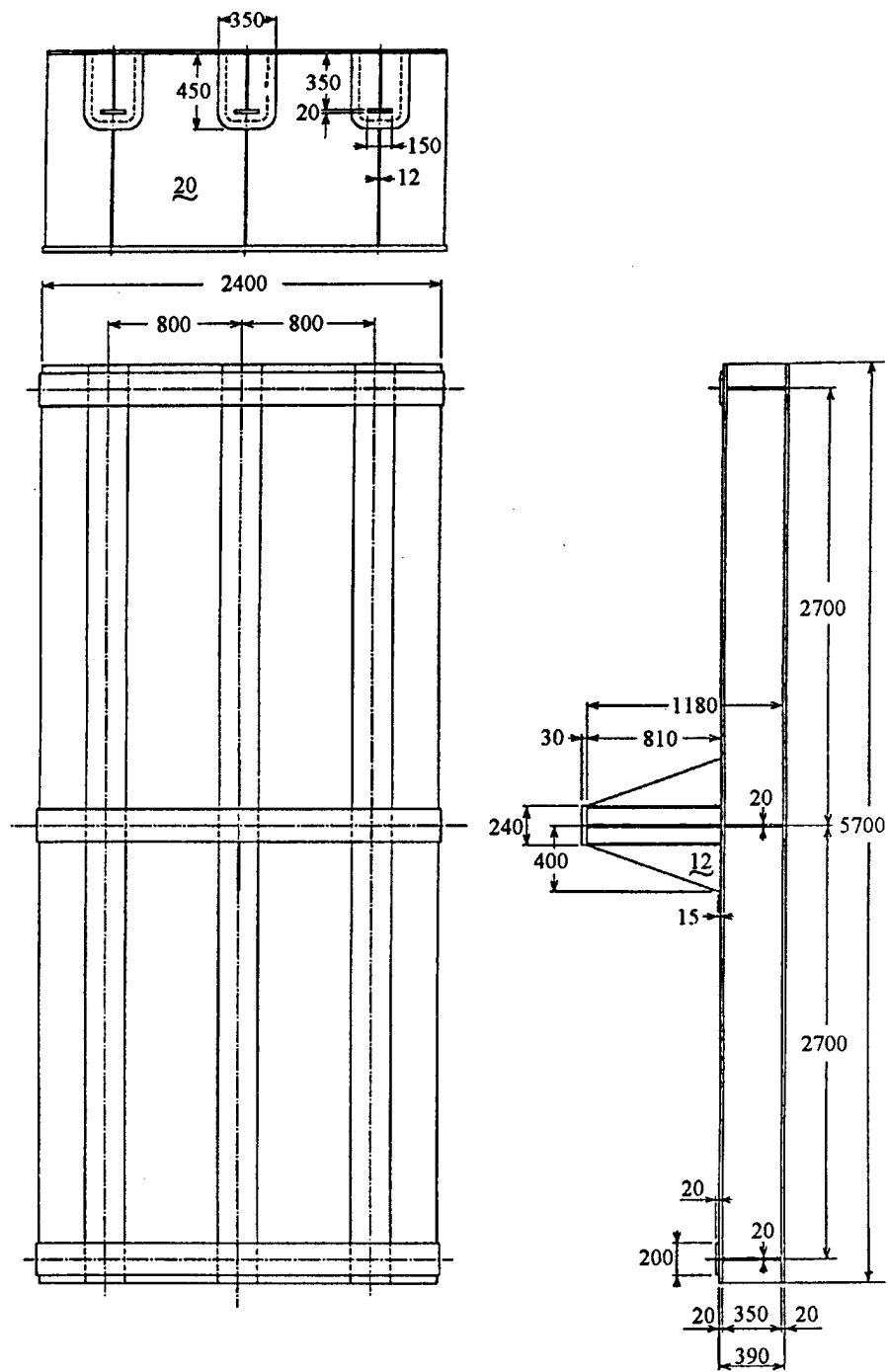


Figure 2: Specimen geometry (in millimeters)

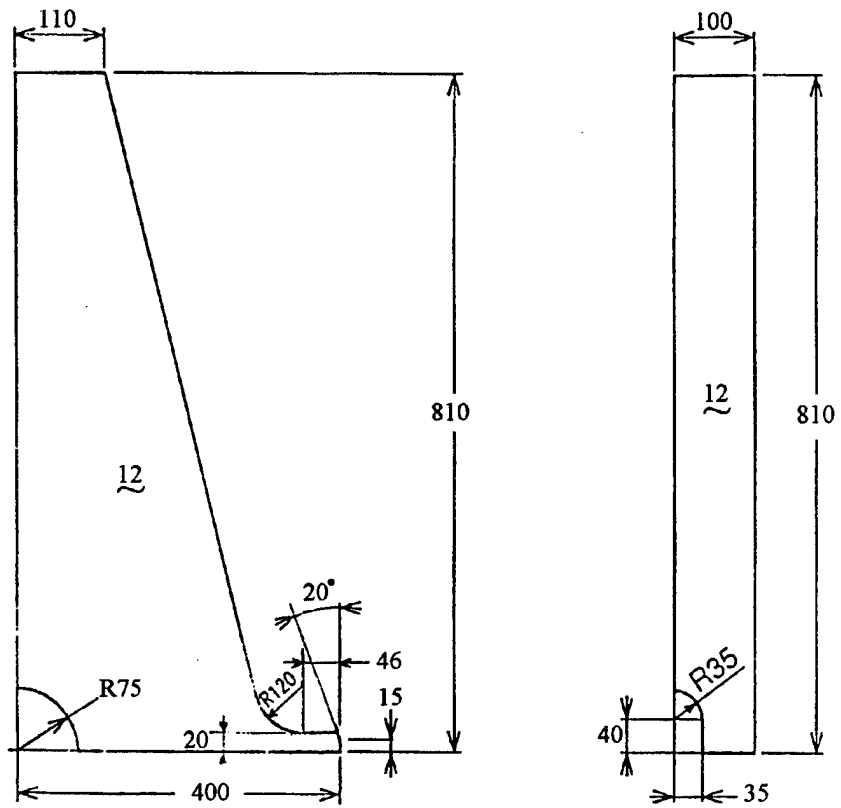
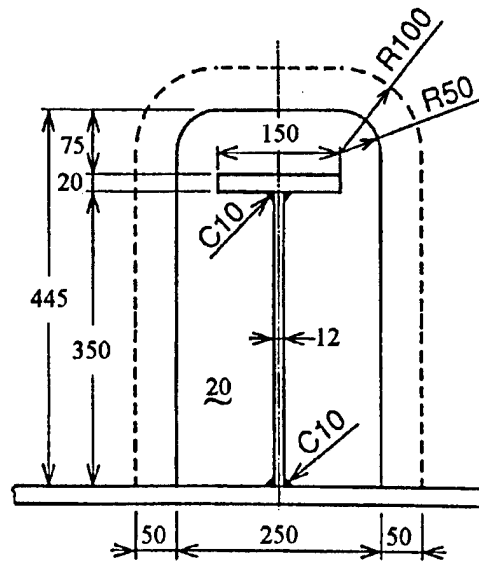


Figure 3: Details of the brackets and collar plate (in millimeters)

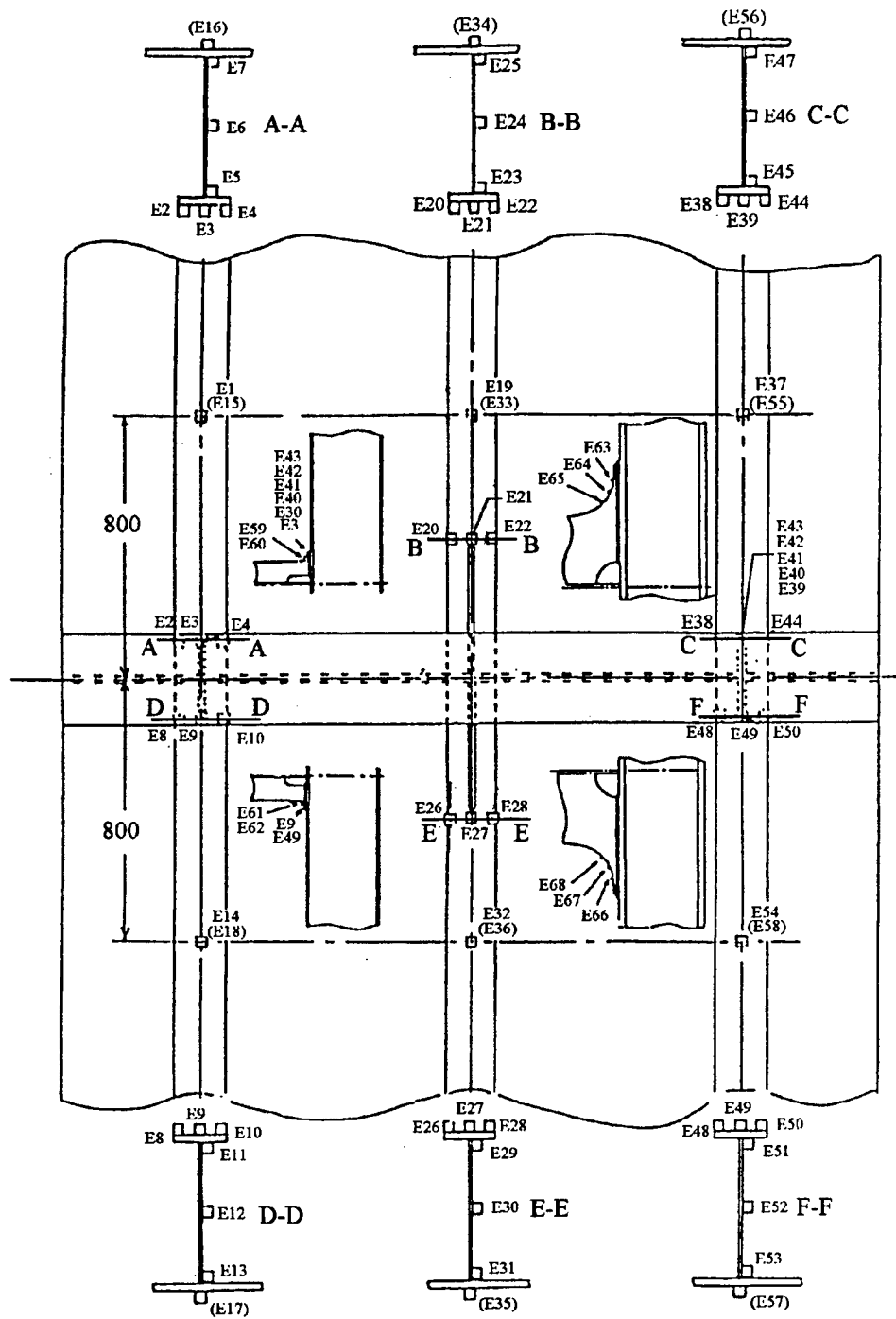


Figure 4: Locations of the strain gauges

4. Finite Element Model

DREA has access to several finite element programs including VAST [9], ADINA [10] and ALGOR [11]. They all have a large selection of element types which can be used to solve a broad spectrum of solid mechanics problems. These programs have a proven accuracy of the results and wide user bases. ALGOR was chosen in this exercise simply because of its availability on the author's PC and the author's familiarity with it. It is capable of performing linear/nonlinear stress, buckling, heat transfer, electrostatic and fluid flow analysis. In addition to the processor, it contains basic pre- and post- processors.

The main area of interest is the weld toe of the side bracket. Since the collar plates at the junction of longitudinals and transverse plate, as shown in Figure 3, are further away from the area of interest, they are neglected in the finite element model. The specimen, therefore, can be considered to be doubly symmetric with respect to the transverse plate and the central longitudinal. This simplification allows only a quarter of the specimen to be modelled. The load in the test was applied through a triangular shape loading plate. To avoid the ambiguity in the modelling of the load, this loading plate is included in the finite element model.

The specimen was modelled with four-node quadrilateral shell elements. This element is formed by superimposing the standard membrane element with the plate bending element [12,13]. The last node of the element can be collapsed into a triangle to form a transition element. For the purpose of easier interpretation of the stress results, it is desired that all the strain gauge locations and the points of interest are coincident with the finite element nodal points. The area of interest is located in front of the weld toe of the side bracket. There are 4 strain gauges and several points of interest at the top surface of the flange which will be used to extrapolate the hot spot stress. The distances between these points are as small as 1 mm. To make the finite element nodes coincident with these points, a very fine mesh is required in this area. The same refined mesh is also needed at the web so that it can be joined together with the mesh on the flange.

The automatic mesh generator in the ALGOR pre-processor CAD system was used to achieve this purpose. The area close to the side bracket was divided into patches. These patches had outlines containing points coincident with the strain gauge locations and the points of interest. These patches were generated separately with an automatic mesh generator and then merged together. Some triangular elements were used in the transition zones from fine mesh to coarse mesh.

A total of 1904 nodes and 1950 elements were used in the model. The geometry of the model, as shown in Figure 5, is referred to an orthogonal Cartesian coordinate system with the x-axis coincident with the junction of the web of the central longitudinal stiffener and the plate. The transverse bulkhead is in the symmetric plane which passes through the origin and is coincident with the y-z plane. The rotational displacements about the y- and z- axes and the translation displacement along the x-axis of the symmetric plane were restrained. In addition, the displacement at the center of the flange of the transverse stiffener at the end of the plate was restrained in the z-direction. A concentrated load of magnitude equal to 98 kN was applied at the top of the transverse bulkhead.

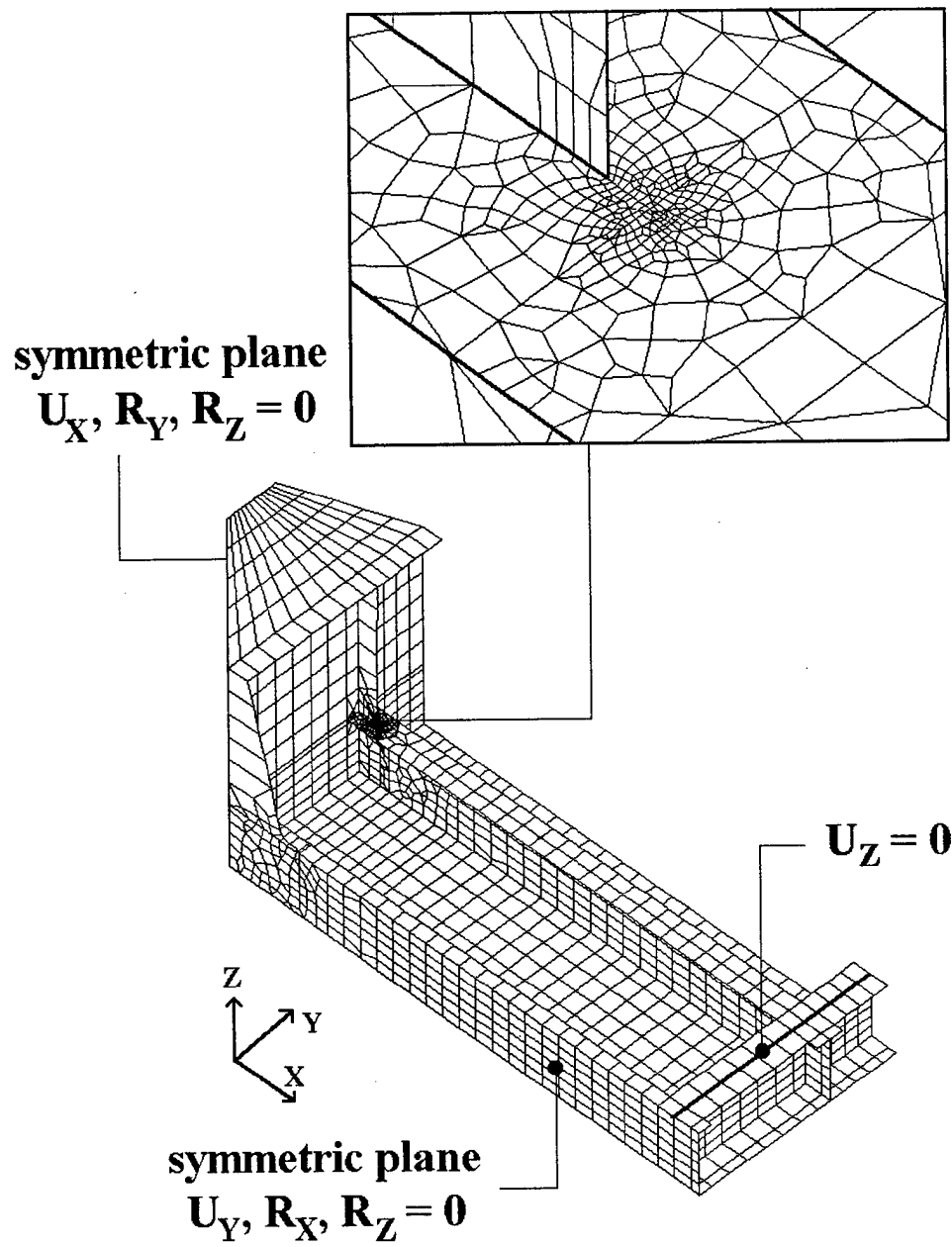


Figure 5: Finite element model

5. Numerical and Analytical Results and Discussion

The finite element method calculates stresses at Gaussian points inside the elements and then uses extrapolation to obtain the stresses at element nodes. For simple structures such as an in-plane loaded plate modeled with plane stress elements or a solid object modeled with solid elements, the corresponding stress components at elements that share the same node can be averaged and be reported in the global coordinate system. However, for complicated structures modelled with sophisticated shell elements, the averaging scheme may not be easy. For shell elements under bending deformation and joined with other elements in an orthogonal direction, the stresses at the top or bottom of the shell cannot be averaged. Like many finite element programs, ALGOR does not report element nodal stresses in the global coordinate system. Instead, the stresses are reported in the local or element coordinate systems.

One of the requirements in this study is to extract the stresses at the top surface of the flange in the direction parallel to the global x-axis (σ_{xx}). In order to compare with the strain gauge results of the test, the combined bending and membrane stresses are needed. A small program was written to convert the reported stresses at the top surface of the shells from the local coordinate system to the global coordinate system and average those that share a common node.

Because of the absence of restraints along the sides, the deformed shape of this stiffened plate is similar to a simply supported beam. The simple beam theory [14] assumes that the cross section of the beam remains planar during deformation. The stresses parallel to the longitudinal axis of the beam can be calculated by

$$\sigma = \frac{M_x y}{I}$$

where x is defined as the distance from one end of the beam; M_x is the moment at location x ; y is the distance from the neutral axis of the cross section; and I is the moment of inertia of the beam. The deflection at location x of the beam subjected to a concentrated load P at the mid-span is

$$\delta_x = \frac{Px}{48EI} (3L^2 - 4x^2) \quad 0 \leq x \leq \frac{L}{2}$$

where E is the Young's modulus and L is the length of the beam.

Under careful examination, however, there are significant differences between a stiffened plate and a simply supported beam. As the stiffener is bent under lateral loading, the load is transferred to the plate through shear action. Consequently, the plate deflects out of plane progressively less than the stiffener. This phenomenon is called the shear lag effect. The assumption that plane sections remain plane in the simple beam theory is no longer valid. Since not all the plate bends with the stiffener, the concept of the effective breadth is used to explain

the behaviour. A constant stress distribution is assumed in the "effective breadth" of the plate. This "effective breadth" of the plate results in an effective bending stiffness of the plate-stiffener combination which is less than the one calculated with the full width of plate. This in turn means that the deflection should be larger than the one calculated from simple beam theory. Further, the primary stresses closer to the central longitudinal should be larger than those calculated from the simple beam theory while the stresses closer to the side longitudinal should be smaller than those calculated from the simple beam theory except for those in the area of the stress concentration.

For the purpose of checking the initial results, the displacements and stresses calculated with the simple beam theory were also compared with the finite element results. The stresses at the bottom surface of the plate across the locations at $X = 461\text{mm}$ and $X = 1200\text{mm}$ from the mid-span are plotted in Figures 6 and 7 respectively. Both show clearly the shear lag effect.

Figures 8 to 11 show the stresses σ_{xx} across the web of the longitudinals at various locations. The stresses at locations near the brackets are larger than those far away from the brackets. These demonstrate the effect of the brackets. Unlike the simple beam theory, these stresses indicate that the cross sections of the stiffeners are no longer remaining planes near the bracket.

The displacements predicted by the finite element method and simple beam theory are compared graphically in Figures 12 and 13 along the junction of the web and flange of the central and side longitudinals, respectively. The shapes of the deflections are similar, but those predicted by the finite element method are larger than those predicted by the simple beam theory which is evidently caused by the shear lag effect.

The stresses σ_{xx} at the top surface of the flange at the junction of the flange and the web for the central and side longitudinals are plotted in the upper parts in Figure 14 and 15, respectively. The stresses σ_{xx} at the bottom surface of the plate at the junction of the plate and the web for the central and side longitudinals are plotted in the lower parts of the figures. Obviously, the brackets have caused the stress pattern to change dramatically. The stresses increase in front of the weld toe and decrease inside the brackets. The stress concentration caused by the sharp corner of the side bracket is shown in the upper part of Figure 15. The detailed stress plot in front of the weld toe of the side longitudinal is shown in Figure 16. After a steep increase, the stress suddenly drops at the corner. This stress singularity shows the deficiency of the displacement based element and the importance of the mesh subdivision near the area of the stress concentration. To give a complete picture of the behaviour of the plate, the overall deformed shape and the Von Mises stress distribution are plotted in Figure 17.

A summary of the results that will be submitted to the ISSC committee II.1 follows:

ISSC II.1 FE Study

Analysis of Japanese SR219 Experimental Model

Name of Contributor: Thomas Hu, Defence Research Establishment Atlantic, Canada;

Software used:

Solver: ALGOR

Pre-Post processor: ALGOR

Structural Modelling:

Type of element: 4-node and 3-node shell elements based on superimposed membrane element and Veubeke plate element

Total number of elements: 1950

Total number of nodes: 1904

Total number of degrees of freedom: 1904×6

Calculated Results:

1. Deflection (at centre of the model, point "D"): 4.913 mm (*beam theory 4.68 mm*)

2. Stresses at the various strain gauge locations are listed in Table 1:

strain gauge location	F.E. (MPa)	beam theory (MPa)	strain gage location	F.E. (MPa)	beam theory (MPa)
E19	86.84	83.41	E37	78.41	83.41
E33	-28.04	-24.17	E55	-24.46	-25.51
E20	94.04	-99.82	E38	104.9	113.0
E21	108.05	-99.82	E40	114.07	112.91
E22	94.04	-99.82	E44	104.53	113.0
E23	85.25	86.45	E45	90.15	97.87
E24	31.59	34.65	E46	33.02	39.22
E25	-22.04	-17.16	E47	-23.66	-19.42
E34	-36.78	-30.53	E56	-36.48	-34.56

Table 1: stresses at various strain gauge locations

3. Stresses and their corresponding strains at locations near the weld toe on the outside fibre of the flange of the side longitudinal are listed in Table 2:

location X		F.E. stress (MPa)	F.E. strain (10^{-4})
3.0	E39	118.80	5.767
4.0	—	116.43	5.652
5.0	E40	114.07	5.537
7.0	E41	112.44	5.458
9.0	E42	110.30	5.354
10.0	0.5T	109.02	5.292
11.0	E43	107.84	5.235
14.8	$1.57T^{3/4}$	106.00	5.146
20.0	1.0T	102.86	4.993
30.0	1.5T	101.39	4.922
40.0	2.0T	100.45	4.876
46.3	$4.9T^{3/4}$	99.83	4.846
60.0	3.0T	100.16	4.862
x: distance from weld toe in mm (Figure 16) T: flange plate thickness (20 mm) * stresses and strains are the components parallel to the global x-axis			

Table 2: Stresses at locations near the weld toe of the side beam

There are a number of rules for extrapolating the hot spot stress at the weld toe. Reference 15 has summarized some of the rules by which the hot spot stress can be extrapolated from the stresses in Figure 16. The following table shows the results of the extrapolation:

rule	a	b	σ_a	σ_b	hot spot stress
1	0.5T	1.0T	109.02	102.86	115.19
2	0.5T	1.5T	109.02	101.39	112.84
3	0.4T	2.0T	111.37	100.45	114.10
4	1.0T	3.0T	102.86	100.16	104.21
5	4.0 mm	10.0 mm	116.43	109.02	121.36
6	$1.57T^{3/4}$	$4.9T^{3/4}$	106.00	99.83	108.90

T: plate thickness (20 mm)
stress at weld toe: 122.69 MPa

Table 3: Hot spot stresses predicted by various rules

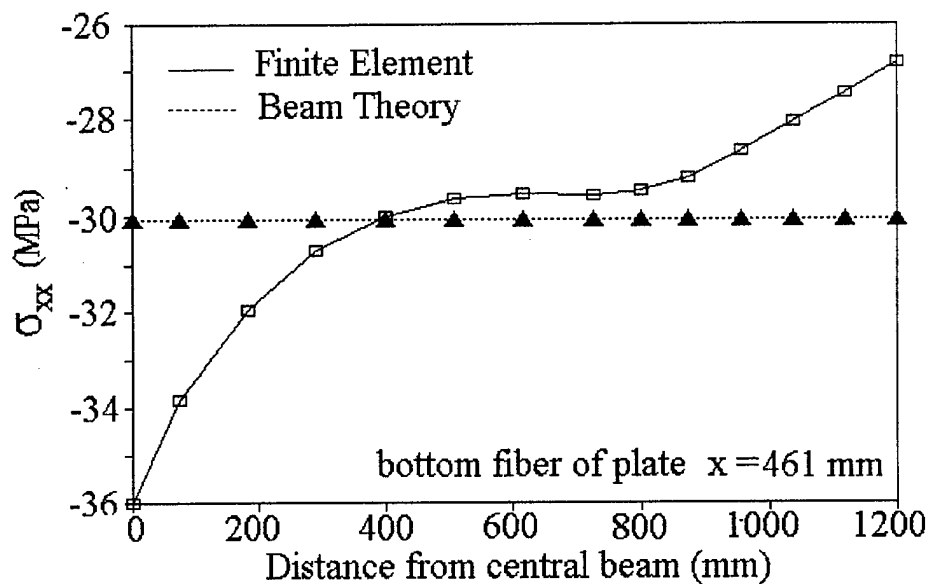


Figure 6: σ_{xx} across the plate at location $X = 461$ mm

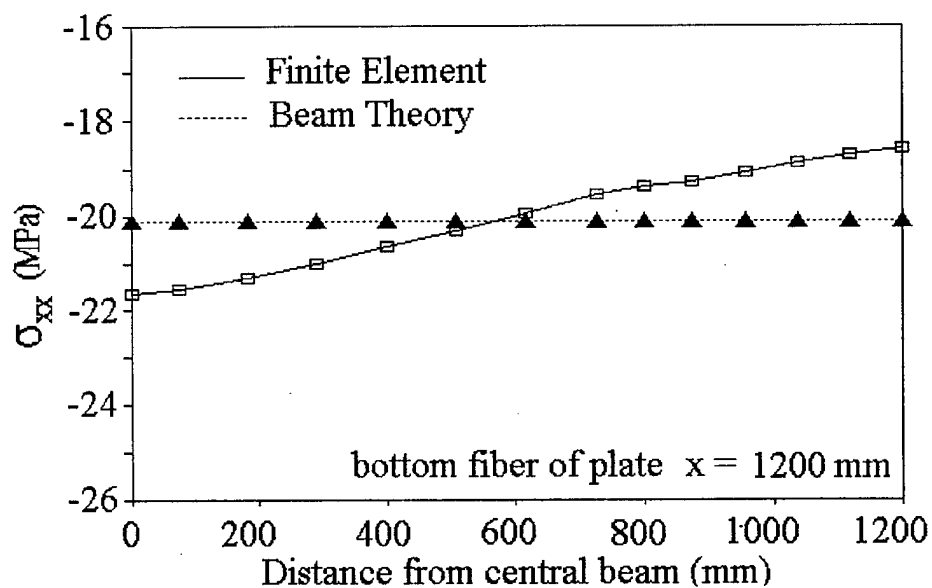


Figure 7: σ_{xx} across the plate at location $X = 1200$ mm

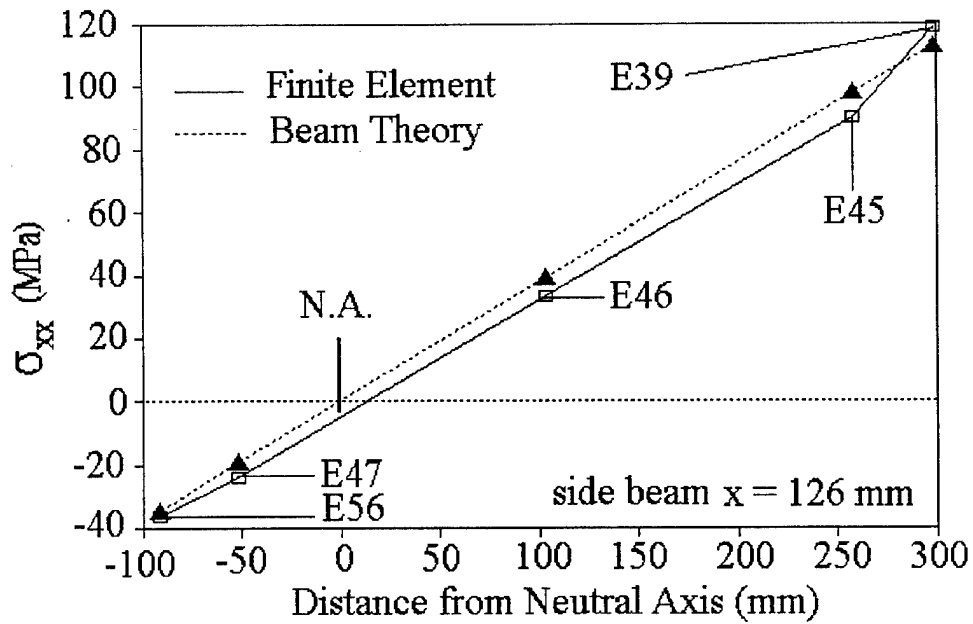


Figure 8: σ_{xx} across the web of the side beam at location near the weld toe

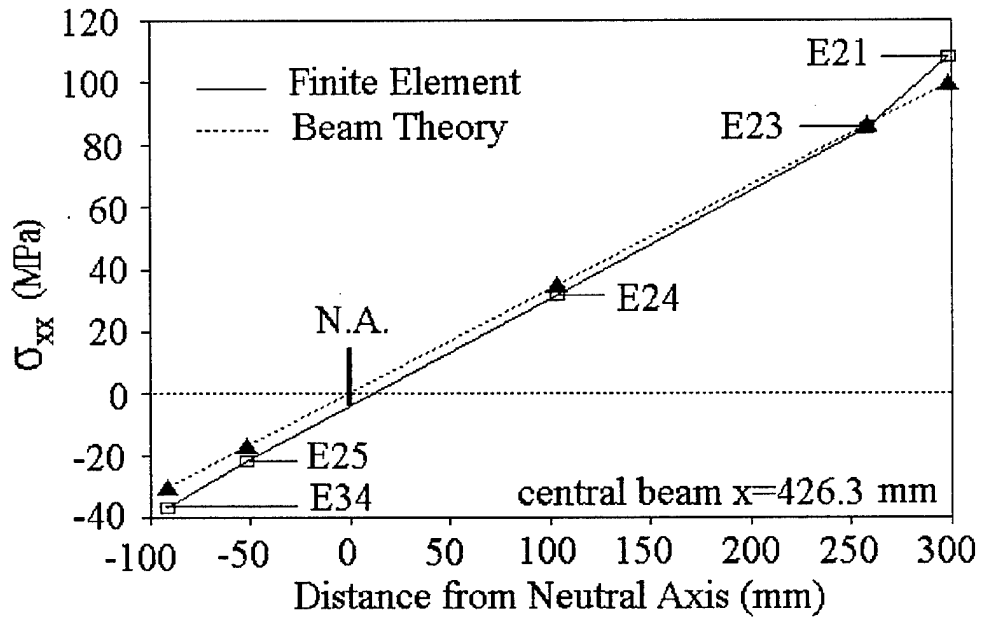


Figure 9: σ_{xx} across the web of the central beam at location near the weld toe

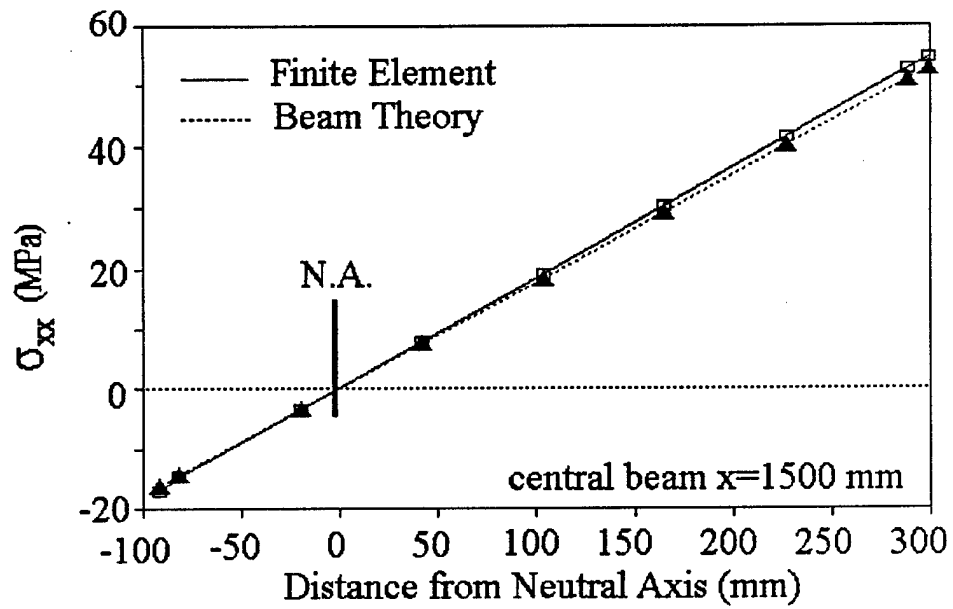


Figure 10: σ_{xx} across the web of the central beam at location X=1500 mm

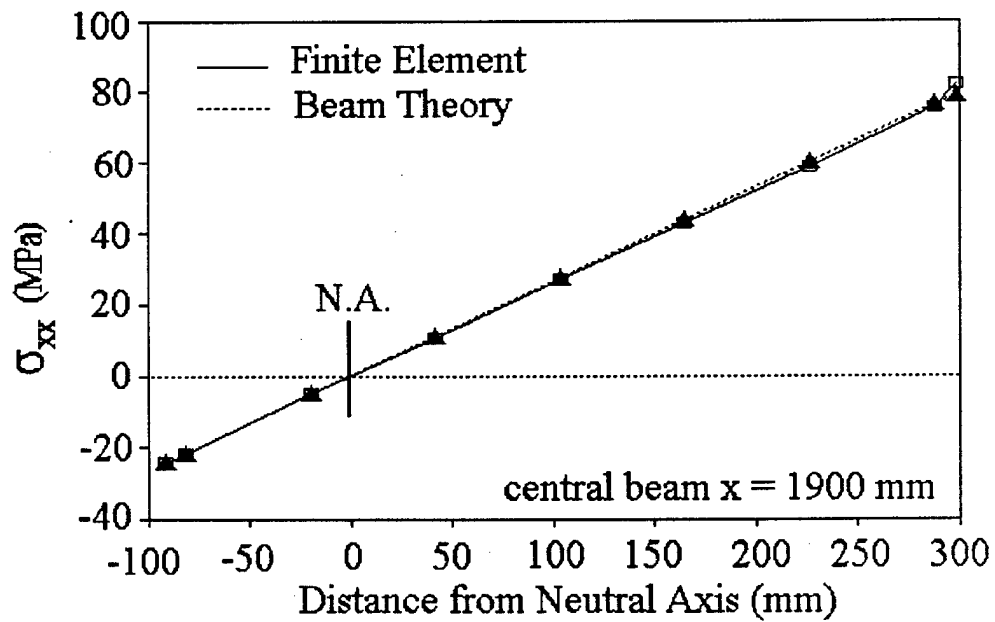


Figure 11: σ_{xx} across the web of the central beam at location X = 1900 mm

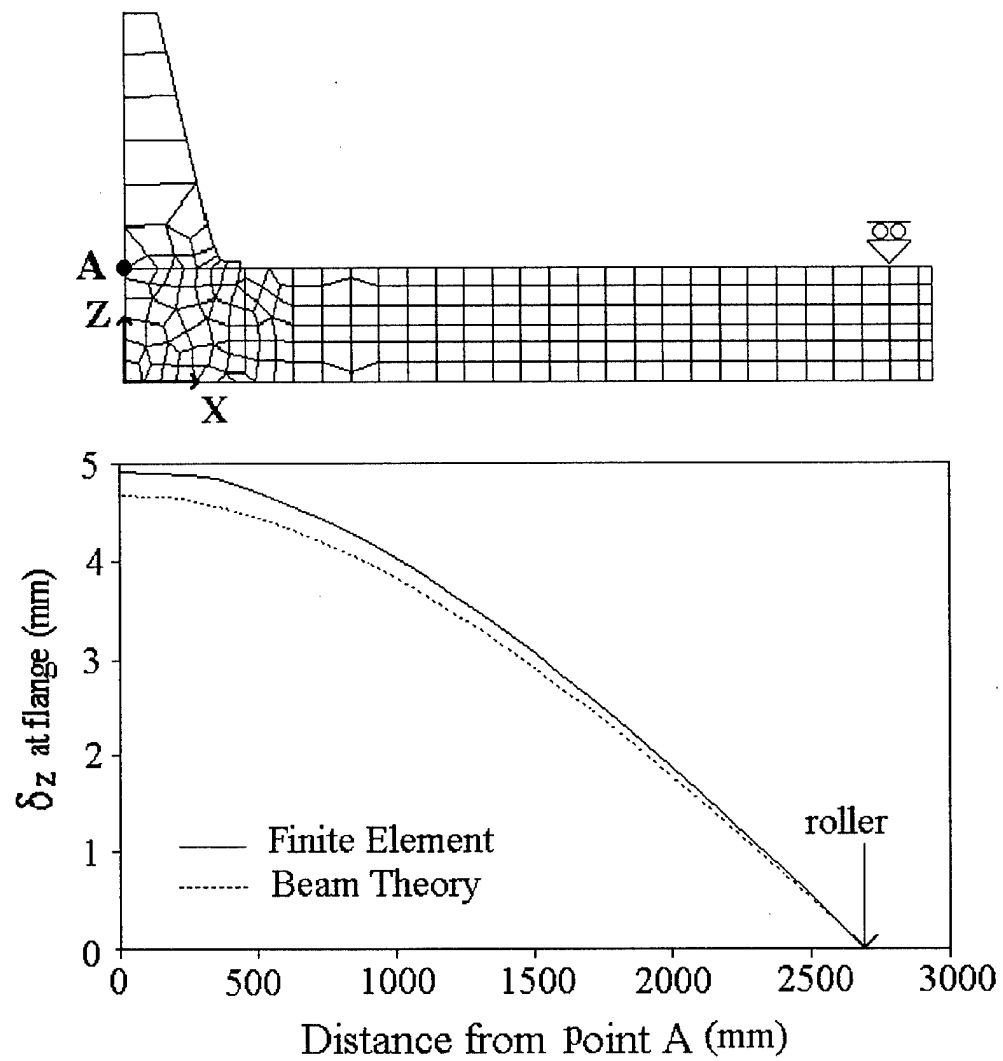


Figure 12: Displacements along the flange of the central beam

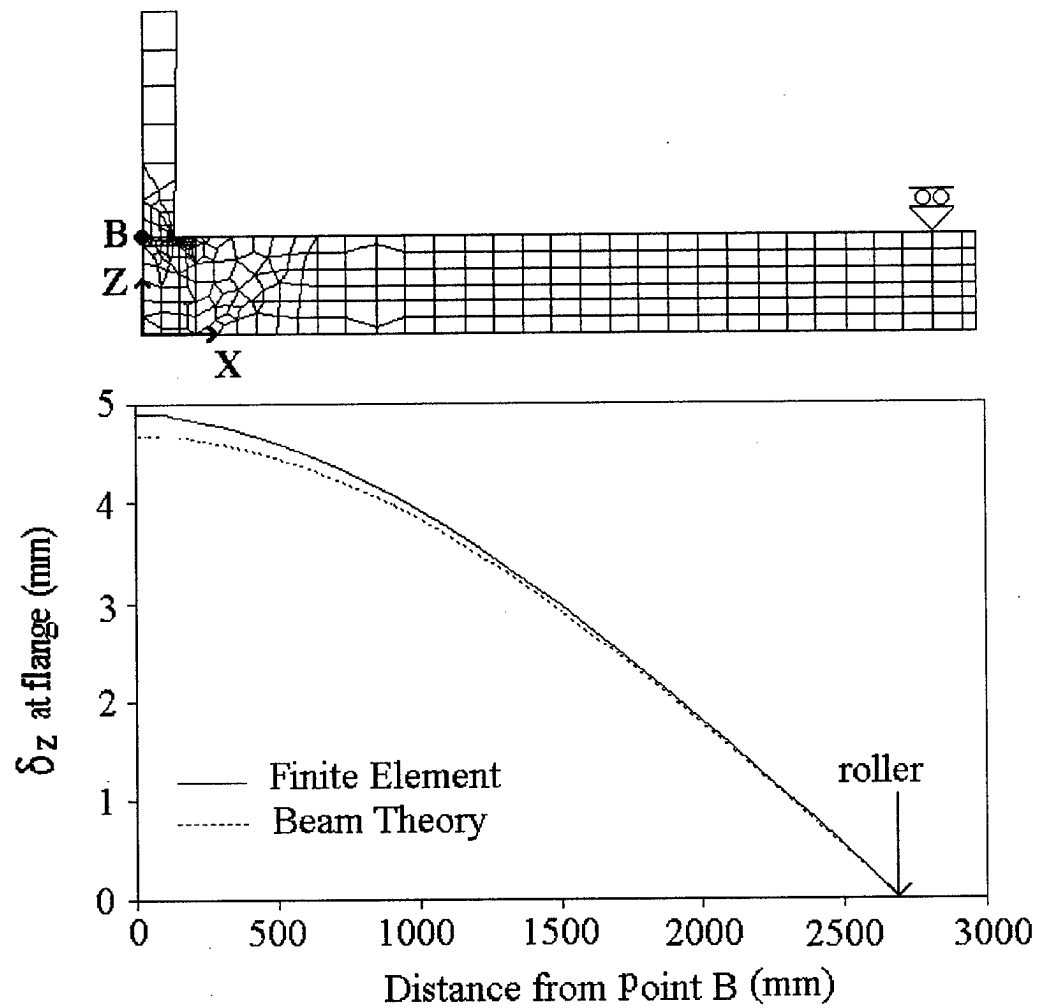


Figure 13: Displacements along the flange of the side beam

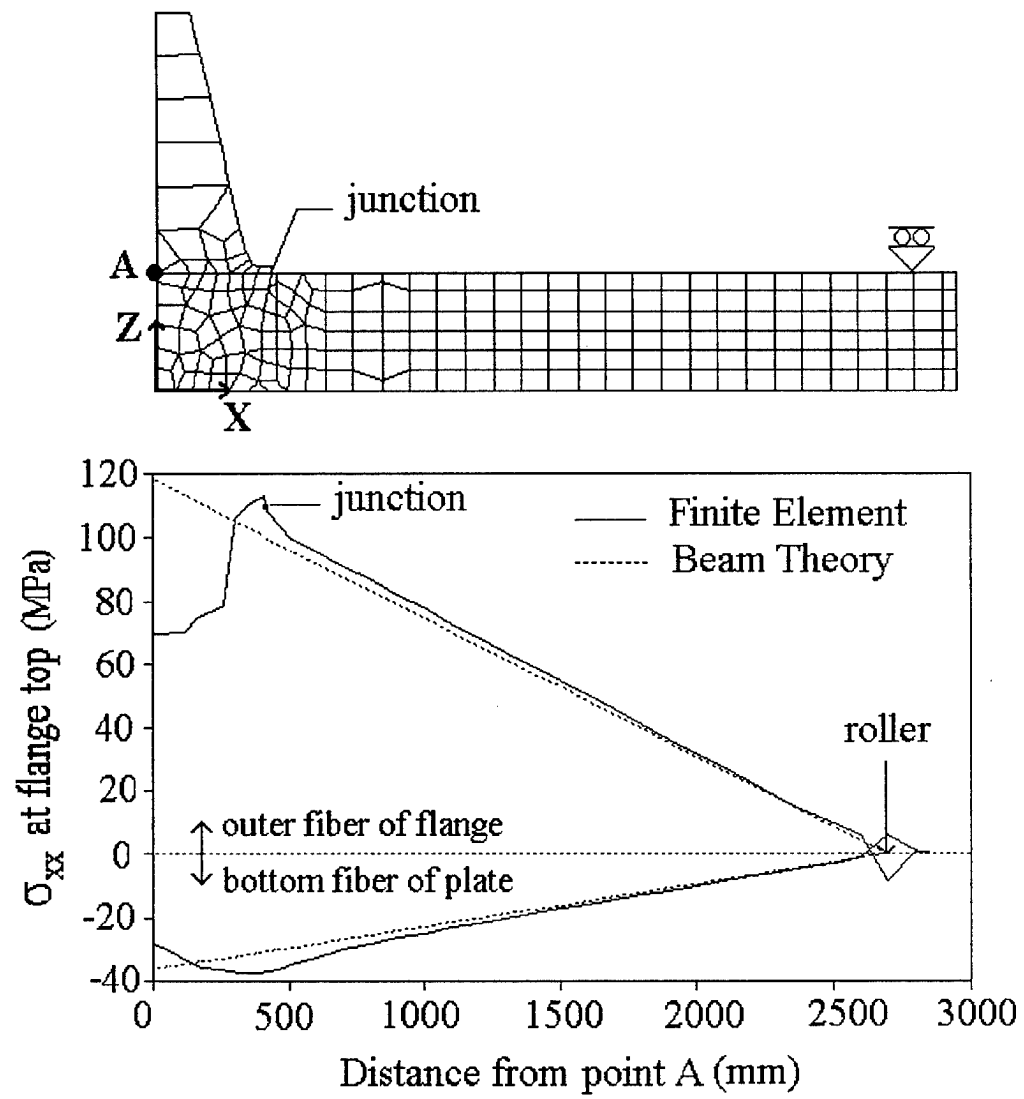


Figure 14: σ_{xx} of the central beam at the farthest fibers from the neutral axis

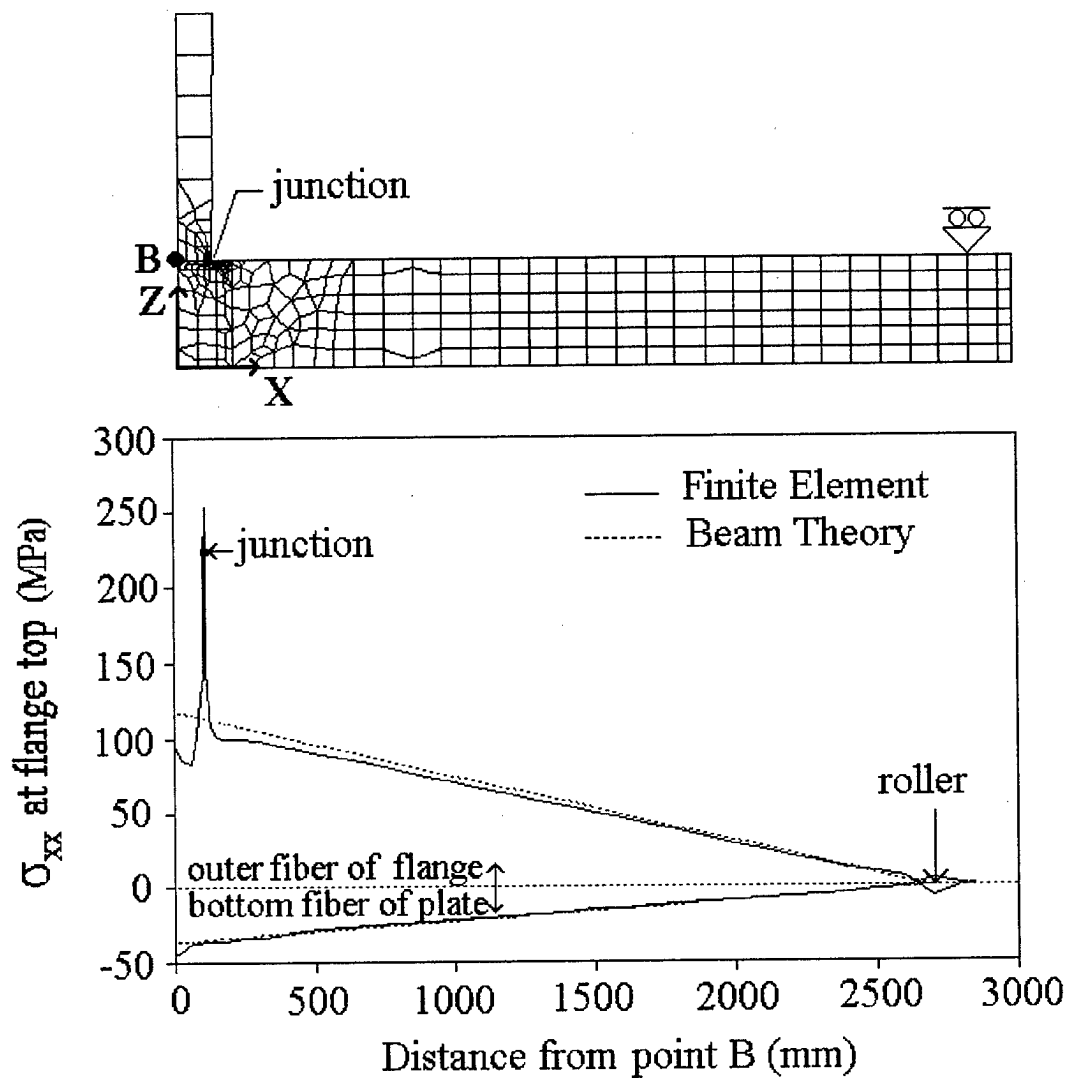


Figure 15: σ_{xx} of the side beam at the farthest fibers from the neutral axis

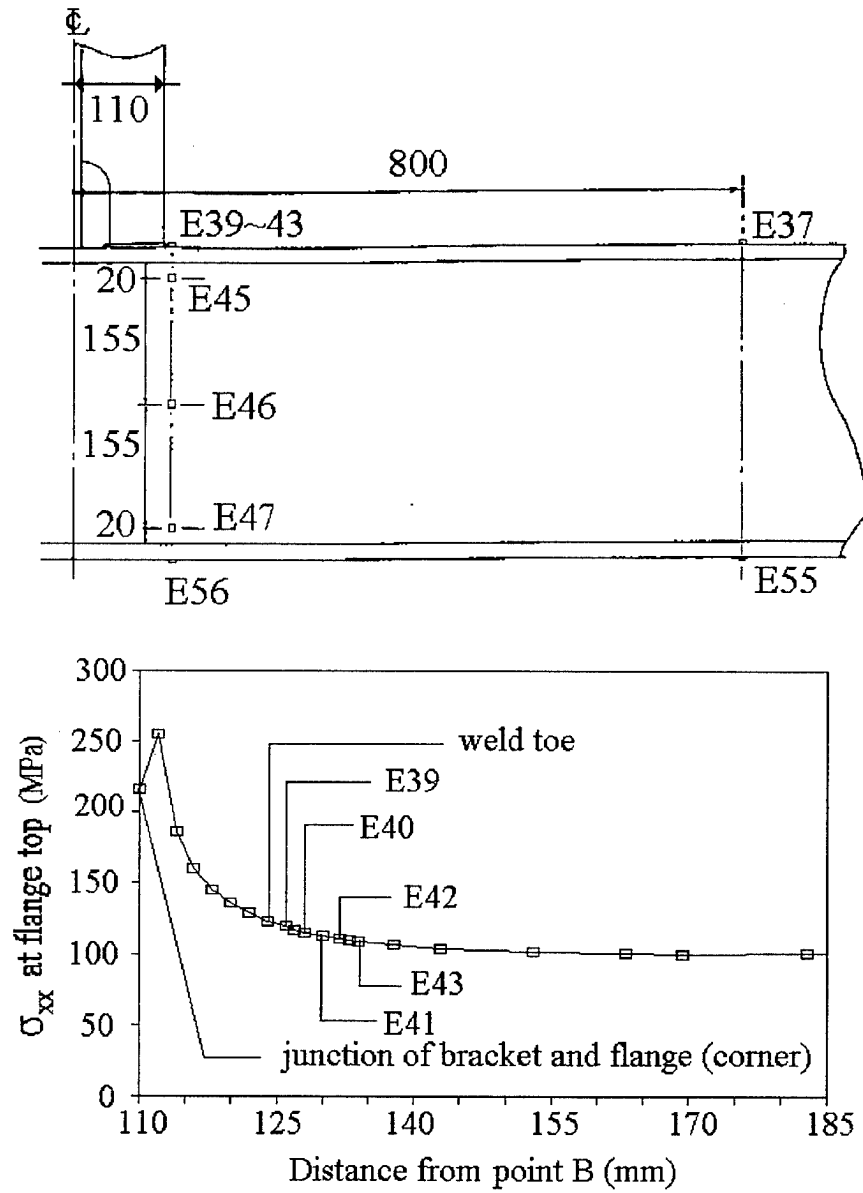


Figure 16: σ_{xx} near the weld toe of the bracket of the side beam

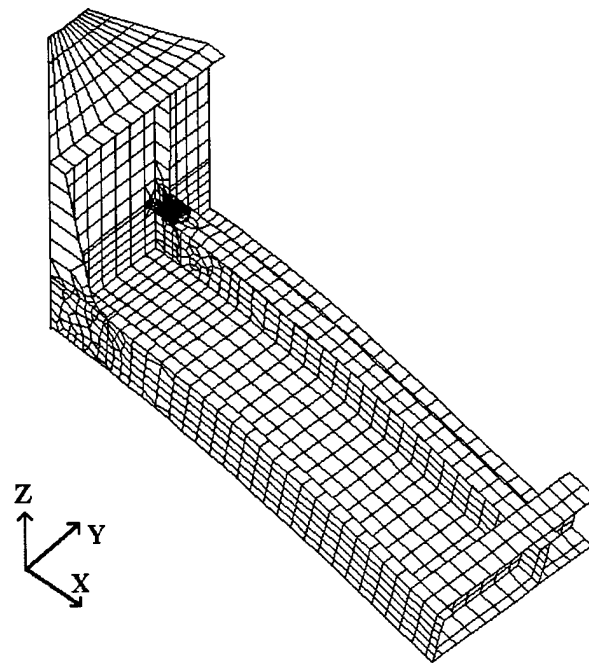


Figure 17: Deformed shape

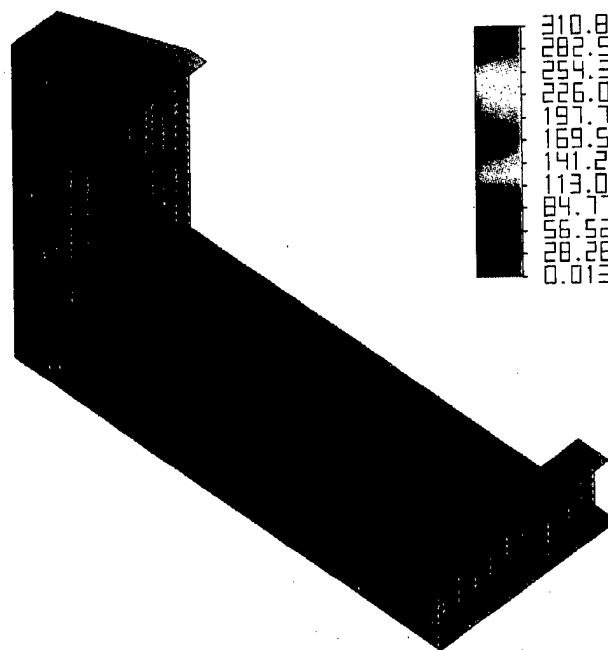


Figure 18: Von Mises stress distribution

6. Concluding Remarks

Algor was found to be a very useful program not only for its solver but also for its pre- and post- processing capability. It can generate the desired mesh with a few mouse clicks. Considering the shear lag effect of the plate and the assumptions behind the simple beam theory, the results obtained from ALGOR seem reasonable. Of course, the test results have not been released at this time. The author believes that the finite element results should be in good agreement with the test results provided that the information about test set-up and procedure is reliable. A future analysis using DREA program VAST to solve the same problem is planned. The results can be used to verify the accuracy and compare the solution time.

The author found that a user's knowledge in applying a specific software package is essential in finite element modelling. Different programs have different algorithms. Users can only learn the tricks through practice and experience. An equally important aspect in finite element analysis is the interpretation of the results. A pretty three dimensional colour plot may give an excellent visual effect but may also give the uninitiated a false impression that the results are always correct.

Although the stresses derived from the finite element method may provide the designer with the means to calculate the hot spot stress, the required level of the analysis is still under investigation. It is generally agreed that a global analysis can only provide the nominal stress. The question is "should the detailed analysis of the local stresses include the profile of the weld shape and size". Some researchers say that the hot spot stress should be invariant for a given geometry. Others argue, based on experimental results, that improved weld profiles can give several times longer fatigue life. Until these disputes are resolved, the designers should always be careful and perhaps use a more conservative criteria.

Fatigue cracking is a common threat to ship structural integrity. The ultimate solution to avoid cracking is to lower stress levels in the areas of possible stress concentrations. A proper detail design can decrease the magnitude of the stress concentration to the point where fatigue crack initiation and growth are either precluded or are deferred to much later in the ship's life thus reducing the risk of crack initiation. The finite element method can be used to study the suitability of the detailed design of the brackets so that the structural performance can be improved.

Users should understand that most finite elements in structural mechanics are based on an assumed displacement field, i.e., the displacement in an element is properly described by the shape function. Stress gradients become steep at the area of the stress concentration. Ordinary shape functions can not be used to describe the rapid change of the stresses. A singularity occurs in the results close to a crack tip. When working in the vicinity of a notch i.e. stress singularity, some care must be taken in the stress analysis. An appropriate finite element idealization is essential to determine the correct stresses.

References

1. "Proceedings of the 12th Ship & Offshore Structures Congress - ISSC94", Edited by N.E. Jeffrey and A.M. Kendrick, Vol. 1 and Vol. 2, St. John's, Canada, 1994.
2. "Minutes of ISSC Committee II.1 Meeting", Washington, D.C., U.S.A., March 28-29, 1995.
3. "Finite Element Comparative Study of Ship Structural Details", for ISSC'97 Technical Committee II.1, prepared by Y. Iwahashi, June 15, 1995.
4. Schulte-Strathhaus, R., "Fatigue Database Development and Analysis", Report No. SMP-1-1, University of California at Berkeley, Department of Naval Architecture, June 1991.
5. "Structural Welding Code - Steel", 13th ed., ANSI/AWS D1.1-92, American Welding Society, Miami, USA, 1992.
6. "Code of Practice for Fatigue Design and Assessment of Steel Structures", BS 7608 : 1993, British Standards Institution, London, 1993.
7. "API Recommended Practice 2A-LRFD (RP 2A-LRFD)", 1st Edition, December 15, 1989.
8. Marshall P.W., "API Provisions for SCF, S-N and Size-Profile Effects", 25th Offshore Technology Conference, Houston, Texas, U.S.A., May, 1993.
9. "Vibration and Strength Analysis Program (VAST): User's Manual - Version 06", Martec Limited, Halifax, N.S., 1990.
10. "ADINA-IN for ADINA Users Manual", Report ARD 90-4, ADINA R & D, Inc., September 1990.
11. "Algor - ViziCad", Vol. 1 & 2, Algor Interactive Systems, Inc., August, 1990.
12. de Veubeke, B.F. and Sander, G. "An Equilibrium Model for Plate Bending", International Journal of Solids and Structures, Vol. 4, pp. 447-468, 1968.
13. de Veubeke, B.F. "A Conforming Finite Element for Plate Bending", International Journal of Solids and Structures, Vol. 4, pp. 95-108, 1968.
14. Gere, J.M. and Timoshenko, S., "Mechanics of Materials", Wadsworth Inc., Belmont, California, 1994.
15. Yagi, J., Machida, S., and Tomita, Y, "Definition of Hot Spot Stress in Welded Plate Type Structure for Fatigue Assessment", J. of Society of Naval Architecture of Japan, Vol 169, May 1991, pp 311-318.

UNCLASSIFIED

SECURITY CLASSIFICATION OF FORM
(highest classification of Title, Abstract, Keywords)

DOCUMENT CONTROL DATA (Security classification of title, body of abstract and indexing annotation must be entered when the overall document is classified)		
1. ORIGINATOR (The name and address of the organization preparing the document. Organizations for whom the document was prepared, e.g. Establishment sponsoring a contractor's report, or tasking agency, are entered in section 8.) Defence Research Establishment Atlantic P.O. Box 1012, Dartmouth, N.S. B2Y 3Z7	2. SECURITY CLASSIFICATION (Overall security of the document including special warning terms if applicable.) <div style="text-align: center; font-size: 1.2em; font-weight: bold;">UNCLASSIFIED</div>	
3. TITLE (The complete document title as indicated on the title page. Its classification should be indicated by the appropriate abbreviation (S,C,R or U) in parentheses after the title.) Hot Spot Stress Analysis of a Fatigue Specimen Using Finite Element Method		
4. AUTHORS (Last name, first name, middle initial. If military, show rank, e.g. Doe, Maj. John E.) HU, Thomas S. Z.		
5. DATE OF PUBLICATION (Month and year of publication of document.) February 1997	6a. NO. OF PAGES (Total containing information. Include Annexes, Appendices, etc.) 35	6b. NO. OF REFS. (Total cited in document.) 15
6. DESCRIPTIVE NOTES (The category of the document, e.g. technical report, technical note or memorandum. If appropriate, enter the type of report, e.g. interim, progress, summary, annual or final. Give the inclusive dates when a specific reporting period is covered.) Technical Memorandum		
8. SPONSORING ACTIVITY (The name of the department project office or laboratory sponsoring the research and development. include the address.) Defence Research Establishment Atlantic P.O. Box 1012, Dartmouth, N.S. B2Y 3Z7		
9a. PROJECT OR GRANT NUMBER (If appropriate, the applicable research and development project or grant number under which the document was written. Please specify whether project or grant.) 1.g.c	9b. CONTRACT NUMBER (If appropriate, the applicable number under which the document was written.)	
10a. ORIGINATOR'S DOCUMENT NUMBER (The official document number by which the document is identified by the originating activity. This number must be unique to this document.) DREA Technical Memorandum 96/232	10b. OTHER DOCUMENT NUMBERS (Any other numbers which may be assigned this document either by the originator or by the sponsor.)	
11. DOCUMENT AVAILABILITY (Any limitations on further dissemination of the document, other than those imposed by security classification) <div style="margin-left: 20px;"><input checked="" type="checkbox"/> Unlimited distribution <input type="checkbox"/> Distribution limited to defence departments and defence contractors; further distribution only as approved <input type="checkbox"/> Distribution limited to defence departments and Canadian defence contractors; further distribution only as approved <input type="checkbox"/> Distribution limited to government departments and agencies; further distribution only as approved <input type="checkbox"/> Distribution limited to defence departments; further distribution only as approved <input type="checkbox"/> Other (please specify):</div>		
12. DOCUMENT ANNOUNCEMENT (Any limitation to the bibliographic announcement of this document. This will normally correspond to the Document Availability (11). However, where further distribution (beyond the audience specified in 11) is possible, a wider announcement audience may be selected.)		

UNCLASSIFIED

SECURITY CLASSIFICATION OF FORM

DDO3 2/06/87

13. ABSTRACT (a brief and factual summary of the document. It may also appear elsewhere in the body of the document itself. It is highly desirable that the abstract of classified documents be unclassified. Each paragraph of the abstract shall begin with an indication of the security classification of the information in the paragraph (unless the document itself is unclassified) represented as (S), (C), (R), or (U). It is not necessary to include here abstracts in both official languages unless the text is bilingual).

DREA is a member of the International Ship & Offshore Structure Congress (ISSC) and currently participates in two of the technical committees (II.1 and III.1). It is recognized that the accuracy of the stress results of a finite element analysis has significant effect in the fatigue life prediction of the structure. Participants of the committee II.1 (the quasi static load effects) have agreed to perform a finite element comparative study. The emphasis of the study is on hot spot stress analysis in which the stress results can be used to determine the fatigue life and the stress concentration factor. In order to draw a quantitative conclusion, a stiffened plate, tested in Japan, was selected to be the subject of this study. Participants were asked to use their own solution procedures and modelling techniques independently. After presenting the background information on hot spot stress analysis in fatigue life prediction and a brief description of the specimen, this memorandum summarizes the finite element model and the solution procedure used by DREA. Numerical results obtained with the finite element method and analytical results from simple beam theory are presented and discussed.

14. KEYWORDS, DESCRIPTORS or IDENTIFIERS (technically meaningful terms or short phrases that characterize a document and could be helpful in cataloguing the document. They should be selected so that no security classification is required. Identifiers, such as equipment model designation, trade name, military project code name, geographic location may also be included. If possible keywords should be selected from a published thesaurus. e.g. Thesaurus of Engineering and Scientific Terms (TEST) and that thesaurus-identified. If it not possible to select indexing terms which are Unclassified, the classification of each should be indicated as with the title).

Fatigue

Hot spot stress

Finite element method

Stress concentration factor

Crack initiation

Shear lag

Effective breadth

**D
R
E
A**



**C
R
D
A**